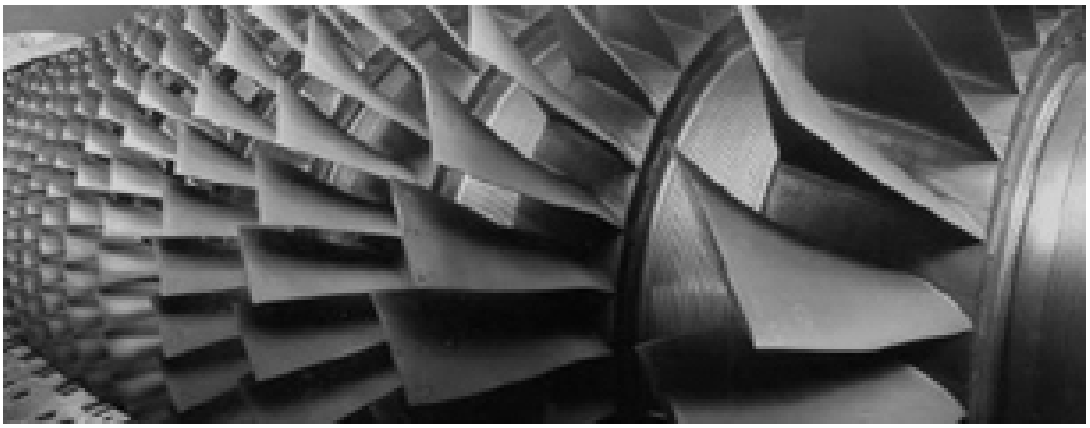
FACULTY OF MECHANICAL
ENGINEERING
UNIVERSITY
OF WEST BOHEMIA

28th Turbomachinery Workshop

Book of Abstracts



October 6th – 8th, 2014

Plzeň
Czech Republic

28th Turbomachinery Workshop

28th Turbomachinery Workshop

Editor: Roman Čermák

Vytištěno jako digitální publikace v PDF

Vydání: 1

Příspěvky ve sborníku jsou vytištěny tak, jak byly zaslány. Nebyla prováděna žádná obsahová ani jazyková korekce. Pokud máte potíže s interpretací textu, kontaktujte přímo autora.

Editor prohlašuje, že tento sborník byl vytvořen s největší možnou zodpovědností. Editor však nemůže garantovat faktickou správnost údajů jednotlivých příspěvků. Čtenářům je proto doporučeno nevyužívat uvedené informace nekriticky, ale konfrontovat je s vlastní profesionální zkušeností.

ISBN 978-80-261-0446-9
Vydala Západočeská univerzita v Plzni, 2014

FACULTY OF MECHANICAL
ENGINEERING
UNIVERSITY
OF WEST BOHEMIA

28th Turbomachinery Workshop

Book of Abstracts

October 6th – 8th, 2014
Plzeň
Czech Republic

evropský sociální fond v ČR
EVROPSKÁ UNIE
MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY
OP Vzdělávání pro konkurenceschopnost
ZÁPADOČESKÁ UNIVERZITA V PLZNI
INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

Workshop byl částečně podpořen projektem CZ.1.07/2.3.00/20.0139
Tento projekt je spolufinancován
Evropským sociálním fondem a státním rozpočtem České republiky

Scientific Committee

Vogt Damian, Prof.Tekn.Dr.
University of Stuttgart, Institute of Thermal Turbomachinery and Machinery Laboratory

Eyb Gerhard, Dr.-Ing.
University of Stuttgart, Institute of Thermal Turbomachinery and Machinery Laboratory

Gampe Uwe, Prof.
TU Dresden

Lampart Piotr, Assoc.Prof.Dr.
Polish Academy of Sciences, Institute of Fluid Flow Machinery, Gdansk

Wroblewski Wlodzimierz, Prof.
Institute of Power Engineering and Turbomachinery, Silesian University of Technology, Gliwice, Poland

Dykas Slawomir, Prof.
Institute of Power Engineering and Turbomachinery, Silesian University of Technology, Gliwice, Poland

Polanský Jiří, Assoc.Prof.
University of West Bohemia, Faculty of Mechanical Engineerin, Plzeň

Šťastný Miroslav, Prof.
University of West Bohemia, Faculty of Mechanical Engineering, Plzeň

Organizing Committee

Čermák Roman, PhD
Gášpár Roman, Ing.
Ratkovská Katarína, Ing.
Cajthamlová Šárka, Ing.
Ajšmanová Naděžda

`

Table of Contents

Modeling and Simulation of the Dynamic Operating Behavior of a High Solar Share Gas Turbine System

Authors: Felsmann C., U. Gampe

Affiliation: TU Dresden, Germany

Keywords: solar, hybrid, gas turbine, power tower, dynamic, control

Summary:

Solar gas turbine systems provide the opportunity to utilize solar heat at a much higher temperature than solar thermal power plants based on steam turbine cycles. Therefore, GT technology has the potential to improve the efficiency of future solar thermal power plants. Nevertheless, to achieve mature technology for commercial application, further development steps are required. Knowledge of the operational behavior of the solar GT system is the basis for the development of the systems control architecture and safety concept. The presentation addresses the basic priciple as well as the dynamic behavior of high solar share GT systems, which are characterized by primary input of solar heat to the GT. To analyze the dynamic operating behavior, a model with parallel arrangement of the combustion chamber and the solar receiver is presented.

Comparison of CFD Flutter Simulations with Measurement Data for a Linear Low-Pressure Turbine Cascade

Authors: Fuhrer C.

Affiliation: Institut für Thermische Strömungsmaschinen Uni Stuttgart, Germany

Keywords: CFD, Flutter

Summary:

Steady measurements on a linear low pressure turbine cascade have been taken out to investigate the flow field. With this data steady CFD simulations are validated. This validated simulation model is used to perform unsteady calculations with the middle blade oscillating in torsion mode. For different reduced frequencies results of an influence coefficient analysis are compared to stability data of unsteady measurements on the test rig. In addition coupled CFD simulations with variable stiffness of the blade suspension are compared to flutter measurement data.

Parametrization of the centrifugal fan segment

Authors: Gášpár R.

Affiliation: University of West Bohemia, Plzeň, Czech Republic

Keywords: Software, parametric model, fan

Summary:

This post deals with the PARAMETRIC fan software. Nowadays the design process of the any turbomachine takes a long time. Especially in case of the optimization problems which are provided by CFD. For CFD processes there is necessary to design a geometry, use a CAD software to create a model, create a mesh, set boundary conditions, calculation itself and evaluation of the calculation results. This software is able to create complex geometry of the impeller segment according to design process rules. It's able to affects almost all parameters of the fan's impeller. The software is also able to parameterize volumes and parts of the segments which are necessary for CFD calculation. This post describes used programming languages, using of the software and its options and limitations.

MEDUSA – a New Way to Control Turbochargers

Authors: Heidinger F., Ilievski M., Müller T., Vogt D.M.

Affiliation: Institut für Thermische Strömungsmaschinen, Uni Stuttgart, Germany

Keywords: Turbocharger, Partial Admission

Summary:

Turbochargers are usually controlled by using a Waste Gate or a Variable Nozzle system (VN). A Waste Gate controlled turbocharger is built with a smaller turbine to achieve under low mass flow rates higher rotational speeds. By exceeding a specific pressure the Waste Gate opens to bypass exhaust gas around the turbine. The energy of the bypassed exhaust gas is lost.
A VN-system uses variable turbine inlet nozzles to control the cross-section area at the turbine inlet. This gives the possibility to control the turbocharger over the whole operation range without wasting energy. The problem for VN controlled TCs are the moving parts of the VN placed in the hot exhaust gas at turbine inlet. The reliability of these parts is low compared to the Waste Gate system because of the high component load of the variable parts.
The proposed paper presents a new control unit based on a partial admission system, which is as well showed in the origin of the name of the system: Multiple Exhaust Duct with Source Adjustment (MEDUSA). The system uses separate controllable flow channels connected to the inlet of the turbocharger turbine. This provides mainly two benefits on the one hand the operation point of the TC can be controlled by closing and opening the flow channels. On the other hand the impulse of the exhaust gas produced by the internal combustion engine is conserved in the flow channels.
This work will presents the MEDUSA system, its function and as well some first measurements.

ORC turbine design

Authors: Klonowicz P., Lampart P., Surwilo J.

Affiliation: Institute of Fluid Flow Machinery, Polish Academy of Sciences in Gdańsk, Poland

Keywords: Organic Rankine Cycle, cogeneration

Summary:

Organic Rankine Cycle (ORC) is a promising technology for small scale cogeneration systems. It offers a possibility to apply low temperature heat sources, allows utilisation of different types of fuels, and also a modular construction which facilitates adaptation of the CHP unit to the required power range. One can think of micro CHP units dedicated for individual households of total heat capacity up to 20kWt and electric power up to 4kWe as well as small CHP modules dedicated for communal energy centres of total heat capacity 500kWt and electric power 100kWe (maximum up to 5 MWt and 1 MWe, respectively).

The main component of an ORC cycle is an expander, usually a turbine working on a special organic/synthetic medium, carefully selected to assure a high efficiency of the thermodynamic cycle.

The paper describes the process of design of blading systems for ORC turbines. Several turbine variants are considered, including an axial flow turbine, radial turbine and radial/axial turbine. The radial and radial/axial turbine have a slightly lower aerodynamic efficiency than the axial turbine, however they consists only of a single stage as compared to the multi-stage axial turbine design.

Fast Calculation of Real Fluid Properties for Steam Turbine CFD Analysis with the new IAPWS Standard on the Spline-Based Table Look-Up Method (SBTL)

Authors: Kunick M., Kretzschmar H.-J., Gampe U., di Mare F.

Affiliation: TU Dresden, Germany

Keywords: Spline-Based Table Look-Up Method, thermodynamic properties, numerical consistency, computational speed, computational fluid dynamics

Summary:

For accurate flow simulations with Computational Fluid Dynamics (CFD) extremely fast algorithms for computing real fluid properties are required. In order to meet these requirements, the International Association for the Properties of Water and Steam (IAPWS) issues the "Guideline on the Fast Calculation of Steam and Water Properties with the Spline-Based Table Look-Up Method (SBTL)" as an international standard. Through the use of the SBTL method, spline-based property functions for the independent variables specific volume and specific internal energy (v,u) are generated for steam and water. Thermodynamic and transport properties, thermodynamic derivatives, and backward functions are calculable in the single-phase, two-phase, and metastable regions. Backward functions of the pressure and specific volume (p,v), as well as specific internal energy and specific entropy (u,s) are calculated with complete numerical consistency to the spline functions from (v,u). The properties calculated from these SBTL functions are in agreement with those of the industrial formulation IAPWS IF97 within a maximum relative deviation of 10 to 100 ppm depending on the property and the range of state. Therefore, the differences between the results of process simulations with the SBTL method and those obtained through the use of IAPWS-IF97 are negligible.

Computations from the (v,u) spline functions are more than 200 times faster than the iterative calculations with the industrial formulation IAPWS IF97.

In order to demonstrate the applicability of the Spline-Based Table Look-up Method the developed algorithms have been implemented into the CFD software TRACE of the German Aerospace Center (DLR). As a result, the computing times for flow simulations of steam turbine stages considering real fluid behavior are reduced by a factor of 10 in comparison to the calculations based on IAPWS-IF97. In comparison to CFD-calculations where steam is considered to be an ideal gas, the computing times are increased by a factor of 1,4 only through the use of the SBTL method.

For generating spline functions for fluid property calculations, the software FluidSplines has been developed. This software enables the application of the Spline-Based Table Look-up Method to all kinds of property functions and to other fluids.

Experimental study of condensing steam flow in nozzles and linear blade cascade

Authors: Majkut M., Dykas S., Strozik M., Smolka K.

Affiliation: Institute of Power Engineering and Turbomachinery,
Silesian University of Technology, Gliwice, Poland

Keywords: wet steam, condensation, experiment, numeric, nozzle, linear cascade

Summary:

Experimental investigations of non-equilibrium spontaneous condensation in transonic steam flow were carried out in nozzles and linear blade cascade. For the tests the geometry of the half arc nozzles were used. The linear cascade consists of the stator blades of the last stage low pressure steam turbine. The applied experimental test section is a part of small scale steam power station located at the Silesian University of Technology. The steam parameters at the test section inlet correspond to the real conditions in low pressure part of steam turbine. The applied linear cascade consists of four stator blades of the last LP stage, it means that the flow through three full blade-to-blade channels were investigated. The information about the flow-filed were acquired by means of static pressure measurements on the pressure and suction side of the one blade-to-blade channel and with the use of Schlieren technique. The static pressure measurements on the blades surfaces were synchronized with the measured total parameters at the inlet and also with the Schlieren pictures, with the frequency of 100Hz. The tests were performed for the wide range of the outlet Mach number (Ma=0.8-1.4).

The capabilities of used measurement techniques were estimated for gaining insight into condensation process in steam flow. The experimental results were compared with numerical calculations carried out by means of an in-house CFD code.

Challenges in the Prediction of Separated Flow in an Exhaust Diffuser

Authors: Mihailowitsch M., Schatz M., Vogt D.M.

Affiliation: Institut für Thermische Strömungsmaschinen Uni Stuttgart, Germany

Keywords: diffuser, CFD, turbulence modeling, validation

Summary:

Flow in a diffuser can comprise complex phenomena such as separation, reattachment, recirculation and shear layer mixing.
Prediction of the flow by means of CFD (Computational Fluid Dynamics) is therefore challenging and strongly dependent on the underlying modeling. In particular the turbulence modeling for such adverse pressure gradient flows is a major field of research and ongoing development.
The presentation aims to give insight into state of the art diffuser CFD, its capabilities and drawbacks.
For this purpose, computational results for a model exhaust diffuser are presented, compared and validated with measurements from an in-house test rig.

Sensitivity Study on Aerodynamic Forcing Prediction in a Highly-Loaded Axial Turbine Stage

Authors: Müller T.R., Vogt D.M.

Affiliation: Institut für Thermische Strömungsmaschinen Uni Stuttgart, Germany

Keywords: aerodynamic forcing, unsteady convergence, scaling, rotor-stator interaction

Summary:

The prevention of aeromechanical problems is an important task in the design process of turbomachines. The interaction between the structure and the surrounding fluid flow can lead to forced vibrations and failures due to High-Cycle Fatigue (HCF). Therefore, a reliable prediction of the aerodynamic forcing is of great importance for all subsequent aeromechanical investigations.

The presented sensitivity study investigates a highly-loaded transonic axial turbine stage. A scaling technique has been applied to modify the stator geometry in order to achieve an integer stator-rotor blade count in an annulus section of the turbine stage. This modification enables a reduction of the numerical effort, while maintaining aerodynamic similarity. A FFT-analysis of the aerodynamic forcing on the rotor has been performed. The aerodynamic forcing on the rotor is mainly dependent on the rotor-stator interaction. Thus, the amplitude of the aerodynamic excitation force at the blade-passing frequency (BPF) has been used as a criterion for the evaluation of the unsteady convergence of the performed transient simulations. The sensitivity of the predicted aerodynamic forcing on the rotor to several important factors such as mesh size, time resolution, transient iteration loops and the applied turbulence model have been investigated within this study and will be discussed.

Modelling unstable operation of compressors

Authors: Polanský J.

Affiliation: University of West Bohemia, Plzeň, Czech Republic

Keywords: CFD, compressor, unstabilities

Summary:

The present study deals with the Integrated Gasification Combined Cycle from a thermodynamics and fluid dynamics point of view.
The paper focuses on the potential risk of fatigue failure of the stator row vanes of an axial compressor in a power plant where low caloric syngas is used as fuel. Dynamically dangerous phenomena can be caused by flow pulsations that were found by numerical as well as experimental investigation of the gas turbine operation. The mechanism of flow unsteadiness was studied using CFD results.
Vortex structures from the cascade at positive incidence angles are responsible for the origin of flow pulsation. The interaction of blade rows plays an important role.
Numerical study of unsteady flow in the cascades of the last compressor stage was carried out by the application of the CFD SW FLUENT ANSYS.
Mathematical model is based on the system of Navier-Stokes equations for the turbulent flow of compressible fluid.
For the turbulence model, the 7-equations Reynolds stress model is adopted. Non-equilibrium wall functions defined in the FLUENT code are used to model the flow near the blade profiles. The numerical model is solved using the Runge-Kutta method in the form of finite volumes.
Coupled implicit scheme with second order accuracy and default under relaxation factors was applied.

Selection of protective coatings for increasing operational life of small jet engine

Authors: [1]Ratkovská K. , [2]Čerňan J., [2]Cúttová M.

Affiliation: [1]University of West Bohemia, Plzeň, Czech Republic
[2]University of Košice, Slovakia

Keywords: small jet engine, protective coatings, thermal plasma spraying, increasing of performance

Summary:

The article deals with the selection of the protective coatings for increasing the performance and operational life of the small jet engine. It contains the structural and chemical analysis of the hot parts of the MPM-20 engine and their operational conditions. Then the appropriate protective coatings in terms of meeting requirements for the surface treatment are selected. Application of the selected coatings (metal and ceramics) on hot engine parts ensure enhancing of service life of the small jet engine and extend its period of operation without the reduction of the nominal power, or other more complex structural modifications.

3D Blade Flutter of the Last Stage Steam Turbine

Authors: Rzadkowski R., Drewcznski M.

Affiliation: Institute of Fluid Flow Machinery, Polish Academy of Sciences in Gdańsk, Poland

Keywords: steam turbine, blade flutter

Summary:

In this study, numerical simulations of 3D viscous and non-viscous flutter of the last stage steam turbine rotor blades were performed.
The developed numerical algorithm solves the 3D Reynolds-averaged Navier-Stokes equation together with Baldwin-Lomax, using the explicit monotonous second-order accurate Godunov-Kolgan finite-volume scheme and moving hybrid H-O structured grid. The structure analysis uses the modal approach and 3D finite element model of the blade. The blade motion is assumed to be a linear combination of modes shapes with the modal coefficients depending on time. The influence of the natural frequencies on the aerodynamic coefficient and aeroelastic coupled oscillations is shown.
The stability (instability) areas for the modes are obtained. It has been shown that interaction between modes plays an important role in the aeroelastic blade response. This interaction has essentially nonlinear character and leads to blade limit cycle oscillations.

Numerical and experimental investigation of a small gas turbine combustor

Authors: Suchocki T., Lampart P.

Affiliation: Institute of Fluid Flow Machinery, Polish Academy of Sciences in Gdańsk, Poland

Keywords: gas turbine, numerical and experimental investigation

Summary:

The paper presents the numerical and experimental investigations of internal combustion in a small gas turbine GTM-140. The non-premixed model of kerosene combustion and discrete phase model with evaporated fuel drops are implemented in the FLUENT solver. The numerical results are validated on the turbine engine test stand. It is equipped with pressure sensors, thermocouples, flow sensors, inductive rotational speed sensor and thrust measuring system. Pressure and temperature sensors are placed equally spaced along the circumference in several parts of the engine: compressor outlet, diffuser outlet, combustion chamber outlet and turbine outlet.

The analysis presents a relationship between values of mass flow rates and pressure as a function of rotational speed. The impact of total pressure losses is discussed. The distribution of holes in the flame tube, which is important for the temperature contours at the inlet to the turbine rotor, is taken into consideration.

Flow analysis of the turbine rotor tip seal on a highly rotary test rig

Authors: Szymański A., Dykas S., Wróblewski W.

Affiliation: Institute of Power Engineering and Turbomachinery,
Silesian University of Technology, Gliwice, Poland

Keywords: CFD, labyrinth, seal, turbomachinery, secondary flow

Summary:

Ever increasing requirements concerning the reduction in the emissions of pollutants into the atmosphere and soil necessitate a rise in the efficiency of power engineering machinery and equipment. Despite the fact that the efficiency value is limited, both manufacturers and scientists make strenuous efforts to improve it. One of the most important factors exerting an impact on turbomachinery efficiency is the type of sealing used in areas separating high and low pressure regions during the relative rotational motion. Sealing bears a 2-3% responsibility for a drop in the gas turbine efficiency. The lower the turbine power output capacity, the bigger the impact of the sealing. Beside state-of-the-art techniques of reducing leakages, such as film riding face seals, finger seals or brush seals, labyrinth seals remain the most common solution used in turbines. They are favoured for their low price, low maintenance costs, resistance to high temperatures and destruction resulting from rubbing (friction on the seal fin-shroud interface). This article presents a detailed CFD analysis of a highly rotary labyrinth tip seal. An impact of the rotor rotational speed and of the pressure ratio on the flow characteristics was observed. The characteristics were calculated for different pressure ratios. Moreover, the effect of roughness was discussed in each case. The calculation results are shown in the form of curves illustrating the mass flow. The conducted analyses show an essential impact of the rotor rotational motion and roughness on the mass flow through the sealing and on the aerodynamic friction factor. Moreover, the analysed profiles of the axial and circumferential velocities strongly depend on the wall roughness. The rotational motion also has a significant effect on the characteristics presented herein. For the rotor rotational speed in the range of 15000 – 24000 rpm, there is a substantial drop in the value of the discharge coefficient from 1 to 0.8 (rough rotor) or 0.87 (smooth rotor); this is accompanied by a rise in the friction factor value. The impact of the rotor rotational motion and roughness is higher for low pressure ratios. For supercritical values of the pressure ratio, the impact of roughness and rotational motion is slight. The conducted CFD analysis confirms the thesis that the discharge coefficient depends on the rotor rotational speed as well as the fact that the flow field is entirely symmetrical in the circumferential direction. For the geometry presented herein, in the case of smooth walls and at the rotational speed value of 24000 rpm, a drop by 12 % was observed in the mass flow. If increased roughness is taken into account, the mass flow – for the highest rotational speed – drops by 20.1% compared to the stationary case. This means that the surface quality affects the flow essentially and that it is possible to exert an impact on the flow characteristics using materials with appropriate porosity.
The optimum working configuration of the sealing can be selected from all those presented in the paper.

Material and life time modelling of a turbo charger casing

Authors: Thiele M.

Affiliation: TU Dresden, Germany

Keywords:

Summary:

not delivered in time

An Overview of the Research at ITSM

Authors: Vogt D. M.

Affiliation: Institut für Thermische Strömungsmaschinen Uni Stuttgart, Germany

Keywords: Research strategy

Summary:

With the arrival of Prof. Tekn. Dr. Damian Vogt as the new director of the Institute of Thermal Turbomachinery and Machinery Laboratory (ITSM) at the University of Stuttgart in August 2013, the research strategy of the institute has been refocused. The presentation will contain an overview of the fields of research currently pursued at ITSM as well as elucidating the methods of research in the various fields.

Dynamic state assessment of the steam micro-turbines with a capacity of 3 kW at operation in CHP ORC system

Authors: Zywica G., Kiciński J.

Affiliation: Institute of Fluid Flow Machinery, Polish Academy of Sciences in Gdańsk, Poland

Keywords: organic Rankin cycle, cogeneration, renewable energy

Summary:

In recent years, with the development of renewable energy sources and distributed cogeneration, more and more popular are systems allow simultaneous generation of heat and electricity in a small scale. One of such technology is the micro CHP ORC system. In the ORC systems, thermal energy extracted from the fuel can be converted into electricity using a steam micro-turbine. Due to the small dimensions, high speeds and a range of operational requirements, micro-turbines are the most advanced component of the micro CHP ORC system.

The article discusses the experimental investigation of the dynamic properties of two alternative variants of steam micro-turbines with a nominal power of 3 kW. Micro-turbines differed solution of the flow system - one of which had 4, and the other only 1 radial stage.

The presented research was conducted in the micro CHP laboratory at IMP PAN in Gdansk. Tested micro-turbines are designed according to the requirements resulting from the characteristics of the realized thermodynamic cycle and a low boiling medium. These tests were carried out with the parameters close to the target operating conditions. These measurements were intended to verify correct operation of the turbo-generator, the evaluation of the dynamic state and the early detection of possible defects. Also the effect of the working environment on durability and reliability of the micro-turbine's components was assessed.

Editor: Roman Čermák
Autor obálky: Roman Čermák
Název publikace: 28th Turbomachinery Workshop
Vydavatel: Západočeská univerzita v Plzni, Univerzitní knihovna
Odd. vydavatelství a tiskových služeb
Univerzitní 8, 306 14 Plzeň
tel.: 377 631 951
e-mail: vydavatel@vyd.zcu.cz

Vydání: 1

Místo vydání: Plzeň
Rok vydání: 2014
ISBN: ISBN 978-80-261-0446-9